SEAMLESS COMPUTING OF FLUID DYNAMICS BY A NODE-BASED FINITE ELEMENT METHOD

G. Yagawa[a], T. Fujisawa[b] and **J. Tsuchida[a]**

[a]Department of Quantum Engineering and Systems Science
The University of Tokyo
Hongo 7-3-1, Bunkyo-ku, Tokyo 113-8656, JAPAN
yagawa@q.t.u-tokyo.ac.jp

[b]Institute of Industrial Science
The University of Tokyo
Komaba 4-6-1, Meguro-ku, Tokyo 153-8505, JAPAN
fujisawa@fsis.iis.u-tokyo.ac.jp

In this presentation, we discuss the *seamless* parallel flow simulation using a node-based finite element method named as the "free mesh method (FMM)" [1-4]. In general, the numerical simulation of fluids requires excessively large calculations so that parallel computing is an indispensable technique. However, usual parallel techniques for CFD focus strictly on the solution of the system of equations, and the mesh generation process is rarely parallelized. Therefore, in parallel environments, remeshing process is the main bottleneck of such computations as fluid simulation with moving boundaries or adaptive capturing of shock waves.

Here, we attempt *seamless* parallel computing involving the mesh generation (i.e. pre-processing) and the solution of the system of equations (i.e. as main-processing) for incompressible and compressible flows on massively parallel processors. The finite elements are generated in the local area around each node and the finite element assembly operations are carried out in a node-by-node manner, where both pre-processing and main-processing are parallelized based on nodes. The amount of the communication among processors is minimized by reordering the nodal identification number using a parallel graph-partitioning library ParMETIS [5]. The concept of finite element mesh in the conventional FEM is represented by a data structure of the connectivity between a central node and associated satellite nodes.

Furthermore, the FMM is featured by a node generation technique, which generates appropriate nodes in the analysis domain based on a probabilistic theory in a parallel manner. Since total processes from pre-processing to main-processing are parallelized with high efficiency, remeshing CFD analysis can easily be conducted on massively parallel processors. The present method is implemented on distributed memory systems such as a PC cluster and a parallel supercomputer Hitachi SR8000. The performance of the method is illustrated by several benchmark problems of incompressible and compressible flows. In addition, a CFD analysis for the pipe organ is shown as an applied example of the present method.

References

[1] G. Yagawa and T. Yamada, "Free mesh method: A new meshless finite element method", *Comp. Mech.*, v.18, p.383-386, 1996.
[2] M. Shirazaki and G. Yagawa, "Large-scale parallel flow analysis based on free mesh method: a virtually meshless method", *Comput. Methods Appl. Mech. Engrg.*, v.174, p.419-431, 1999.
[3] G. Yagawa and T. Furukawa, "Recent developments of free mesh method", *Int. J. Numer. Meth. Engng.*, v.47, p.1419-1443, 2000.
[4] T. Fujisawa, M. Inaba and G. Yagawa, "Parallel computing of high-speed compressible flows using a node-based finite element method", to appear in *Int. J. Numer. Meth. Engng.*, 2003.
[5] METIS URL http://www-users.cs.umn.edu/~karypis/metis/